CURA AND GCODE GENERATION

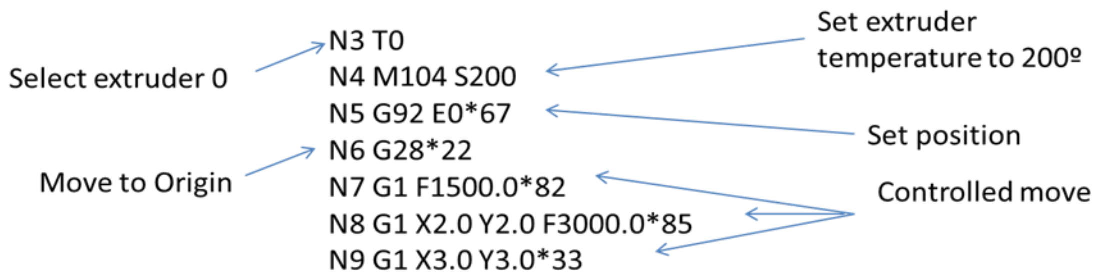
Written by Greta D'Angelo

In this section:

- An introduction to Gcode
- Introduction on Cura and its main functions
- Overview on Basic configuration
- Overview on Advanced configuration
- Printing multiple objects
- Manipulation tools

INTRODUCTION TO GCODE

GCODE is a Numerical Control (NC) programming language used by a wide range of machines in order to be able to perform operations of various kinds as CNC milling machines, Robots and 3Dprinters. Therefore a G-code file appears as a list of instructions that the machine will be able to interpret. For a 3D printer for example these instructions are: speed, path, extrusion temperature, fan, filling density and many others. These are interpreted by the firmware placed in the printer's electronics. In the picture below it is possible to see a typical piece of Gcode.



The list is read top-down, and all the orders are executed consecutively. Gcode is automatically generated by software which in the case of the Ultimaker it is called Cura. However there are also other slicing software available such as Slic3r.

To generate the code first is necessary to upload the model in the software. Subsequently after the parameters are verified and set up the paths are generated (nozzle trajectories for perimeters, infill, support structures; temperature and others). In the end the Gcode is exported and ready to use.

INTRODUCTION TO CURA

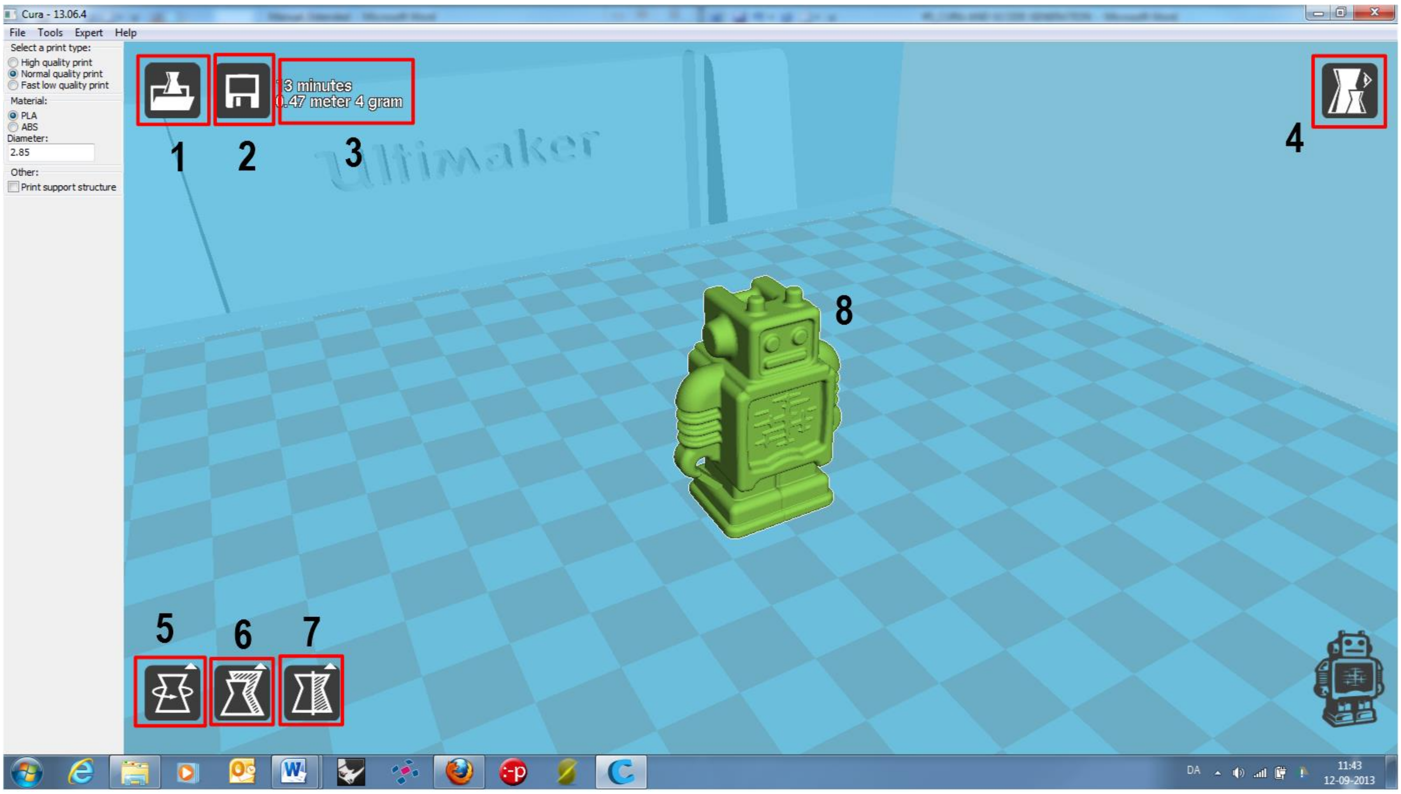
As mentioned Cura is the slicing software used by Ultimaker to prepare the files for printing. It is possible to download for free the latest version (13.06.4) on the official page of the Ultimaker software: http://software.ultimaker.com/ . Windows, Mac and Linux versions are available. Cura is already installed in all the computers of the Fablab and they have already the right parameters for printing.

IMPORTANT: *The use of Fablab's PCs for slicing is highly recommended. They are all fully equipped with the most updated version of Cura and the right parameters.*

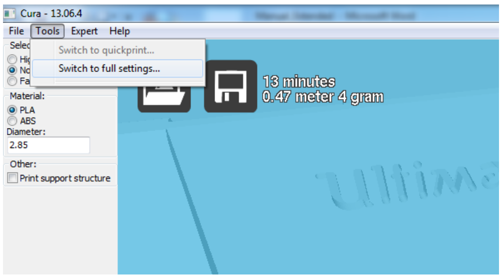
After installed and once opened, the first screen will look like the picture below where it is possible to see some icons and they correspond to:

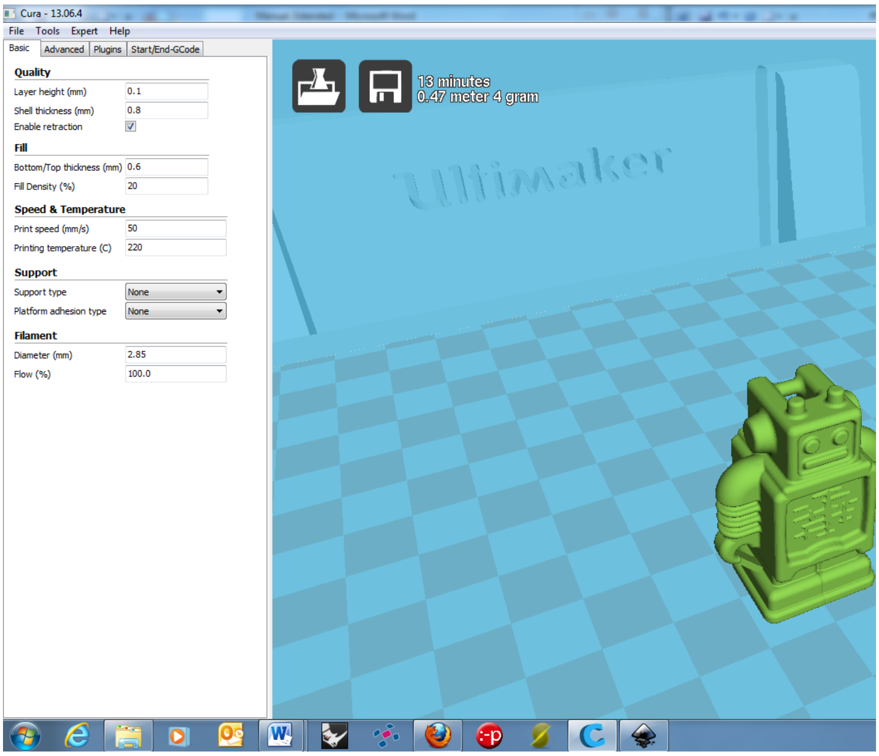
1. Loading a new file

2. Saving the Gcode
3. Estimation on the time required for the print and the amount of material used
4. View mode (normal, Overhangs, transparent, x-ray, Layers)
5. Rotate
6. Scale
7. Mirror
8. Preview of the model on the build platform



Before start doing anything on this set up, remember to switch to the full settings. In this way, the left bar will be replaced by the parameters options that you can change according to needs. To switch to full settings, from top toolbar select: *Tool> switch to full settings* (as shown below). However is necessary to switch to full settings only if using another computer and only after the first installation. Fablab's computers already operate with full settings mode.





Parameters can be changed differently. If you are using the Fablab's computers then you most likely do not need to change anything unless specific requirements have to be fulfilled. In the case a personal computer is used, it is possible to find in the following sections a description of the parameters that can be found and change in the software. There are three places where parameters can be changed at different level:

- Basic tab
- Advanced tab
- Expert tab (top toolbar)

Basic configuration

Quality

Layer height: Controls the thickness of the layers. This directly affects the quality of your final print. The default of 0.2mm gives decent prints, but for high quality prints is also possible to lower the layer thickness to 0.1mm, notice that this will double the building time.

Shell thickness: The thickness of the outside walls. In case of a cube this controls the front, back and side thickness. Normal thickness of 0.8mm (2 perimeters) gives good results, but small prints it's sometimes better to set this to 0.4mm (1 perimeter).

Enable retraction: Retraction pulls back the filament when moving over an open space. This reduces the string effect in prints, but for printing single objects with no different areas, it's not necessary. This option should be actuve.

Fill

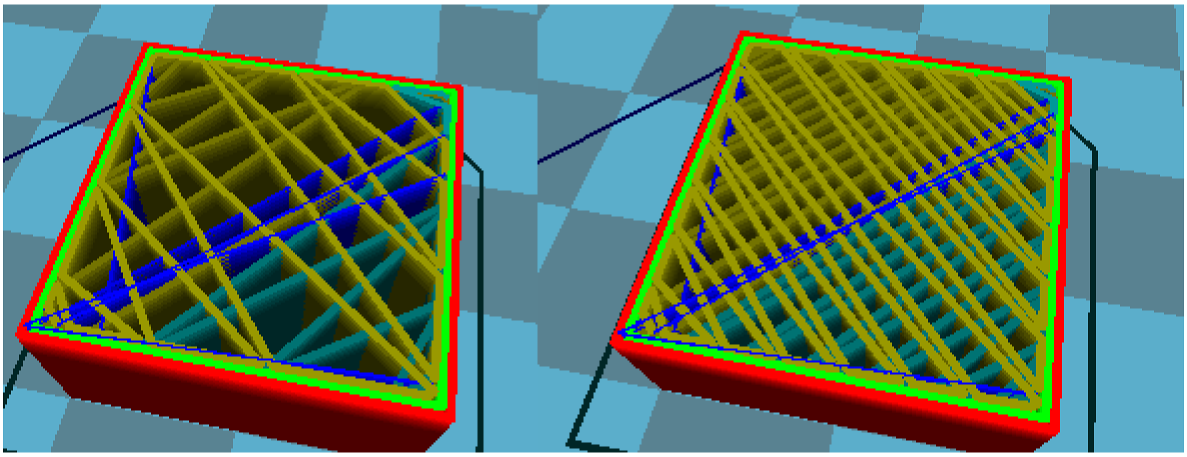
Bottom/Top thickness: The bottom/top thickness controls how many layers are fully filled. Setting this higher gives a stronger part, setting it lower gives a weaker part but it will print faster. In general 0.6mm gives strong parts without holes, going lower can cause holes to appear in at the top of your models.

Fill density: The fill density controls the amount of *sparse infill* that is made. A higher infill is more likely to cause some warping in the part, on top of increasing the printing time. Therefore it is not recommended to use high infill value if not needed. Default value: 20%.

SPEED AND TEMPERATURE

Printing speed: Controls the speed of the printer head when it is extruding. Default value: 20 mm/s. recommended to keep a small value in the software. It will be possible to increase the printing speed directly from the machine (see HOW TO USE AN ULTIMAKER).

Printing temperature: The temperature of the nozzle. The printer head heats up to this temperature before starting the print. 195 degrees Celsius by default.



SUPPORT

Support type: Type of support material. Support material is a structure printed below your print to support areas of the print that would otherwise collapse. It is possible to choose among three configurations

- Exterior only: adds support structures only when they will touch the building platform.
- Everywhere: adds support structures everywhere they are needed, even o the insides of the model
- None: does not add support structures even if they might be needed

Platform Adhesion type: it is possible to choose on how to make the part to stick to the build plate. In most of the case this option is recommended especially when printing large objects. The machine will perform two extra layers, before starting the construction of the part that helps the surface to stick to the plate and to avoid the warping. The options are:

- None: there will be no additional layers before the part. Therefore the first layer to print will be the lower layer of the part
- Brim: the first layers are a series of concentric paths around the object (see picture below). It gives the best result as it does not scare the object.
- Raft: the first layers are crossed paths underneath the object (see picture below)

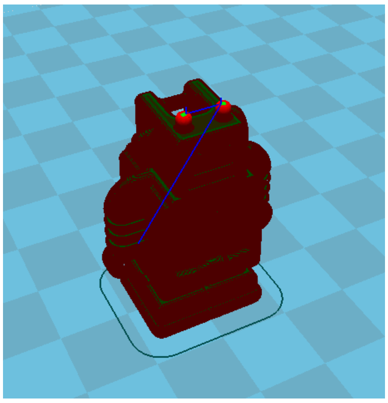


FIGURE 1 NONE

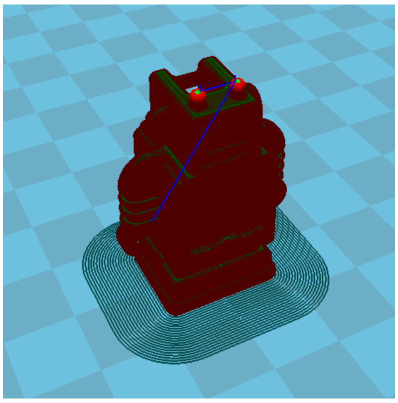


FIGURE 2 BRIM

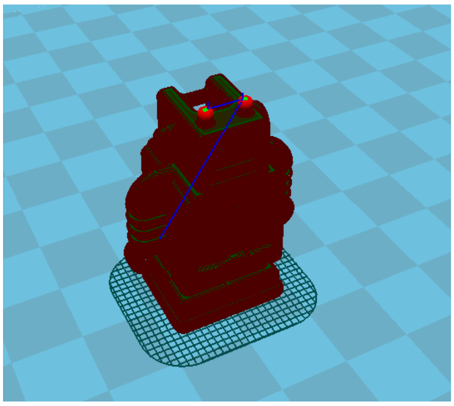


FIGURE 3 RAFT

FILAMENT

Diameter: This is the diameter of the filament in mm. Digital calipers are needed to measure the filament up to 2 decimals. Setting this correctly improves the final print quality. Average 3mm filament has a diameter of 2.89mm.

Flow: flow rate of the molten material.

Advanced Configuration

Machine

Nozzle size (mm): It is the diameter of the nozzle's hole. Do not change this value.

Retraction

This feature is used in order to avoid the ooze of the nozzle when it is crossing the perimeters during the travel movements.

Speed: Speed at which the filament is retracted. Default value of 40 is good, lower values can make retraction to take too much time, and more speed could damage the filament grinding it.

Distance: Amount of material retracted, values from 2 to 4 are good. If 0, retraction is disabled.

Quality

Initial layer thickness (mm): Layer thickness of the bottom layer. A thicker first layer makes sticking to the bed easier. Set to 0 to get the same layer height as the others.

Cut off object bottom (mm): this can be used for objects that do not have a flat bottom and this creates a small initial layer. Remember that a first layer big enough is really important, especially if printing without support structures.

Dual extrusion overlap: This parameter is to use only in case the machine is set up for a double extrusion, which in the Fablab case the machines are not. Therefore ignore this command.

Speed

Travel speed (mm/s): Speed of the machine when travelling. Default value of 150mm/s.

Bottom Layer Speed (mm/s): Print speed for the bottom layer. Low speeds help the first layer to stick better. 15mm/s is the default value.

Infill speed: this is the speed that the nozzle has when filling the perimeter.

Cool

Minimal layer time (sec): minimum time spent in a layer, gives the layer time to cool down before the next layer is put on top. This feature slows down the printing speed of small layers to make sure they spent this amount of seconds.

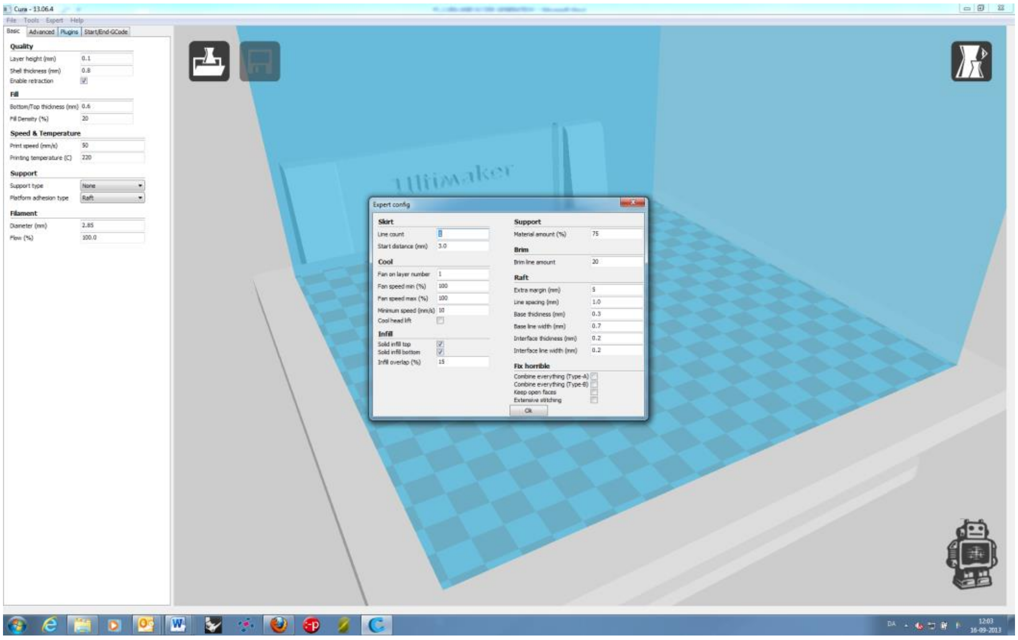
Enable cooling fan: Enable the cooling fan during the print. It should be always enabled.

Expert configuration

It is possible to navigate in the Expert settings from the top tool bar> Expert> Open Expert settings. A dialogue window as shown in the picture below will appear. Changing this settings is not really recommended unless in the need of something extremely specific. In that case contact one of the technicians. However there are two features that might be useful in some cases. They are:

- Solid infill Top
- Solid infill Bottom

If the boxes are unchecked the part will be printed without top and without bottom. Also it can be decided to uncheck only one of the two.



PRINTING MORE THAN ONE OBJECT AT THE SAME TIME

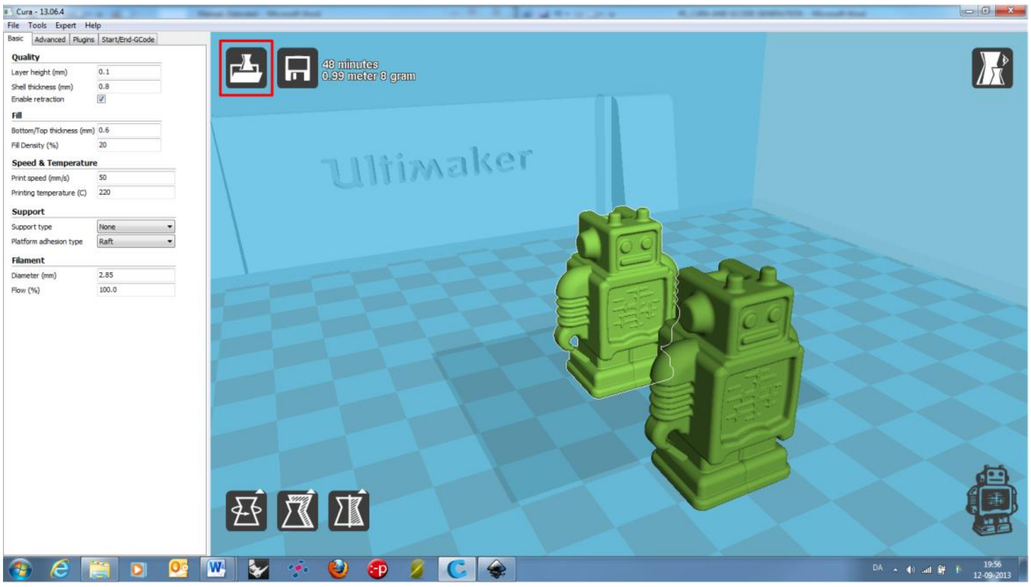
WARNING: make sure that when multiple objects are printed this is done not one object at the time but all the objects at the same time. In doubt ask a technician.

There are two ways to print more than one object in the same print job:

#1

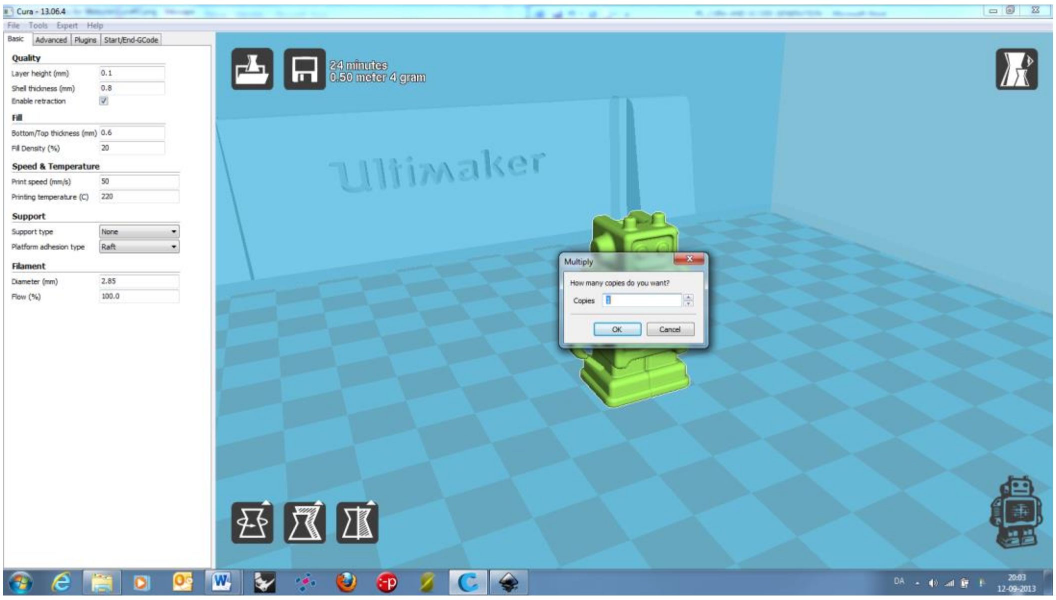
By selecting the *Load* icon in the up left corner and then selecting the additional model to print. After few seconds this will appear on the screen close to the first model. It is possible to rearrange the parts as wanted. However the distance between them is fixed. The machine will print one object at the time.

Also it is possible to load as many models as they fit in the build platform. If not satisfied with the result, it is possible to select individually the parts and delete them by pressing the *delete* button on the keyboard.



#2

If willing to print more of the same parts, hence to duplicate them then: select the part (left click on the mouse), after the part is selected then right click with the mouse. A small dialogue window will appear. Select *Multiply.* Another dialogue window will appear asking how many copies are wanted. Then insert the desired amount and press *Ok.* The copies of the part will appear close to the original one.

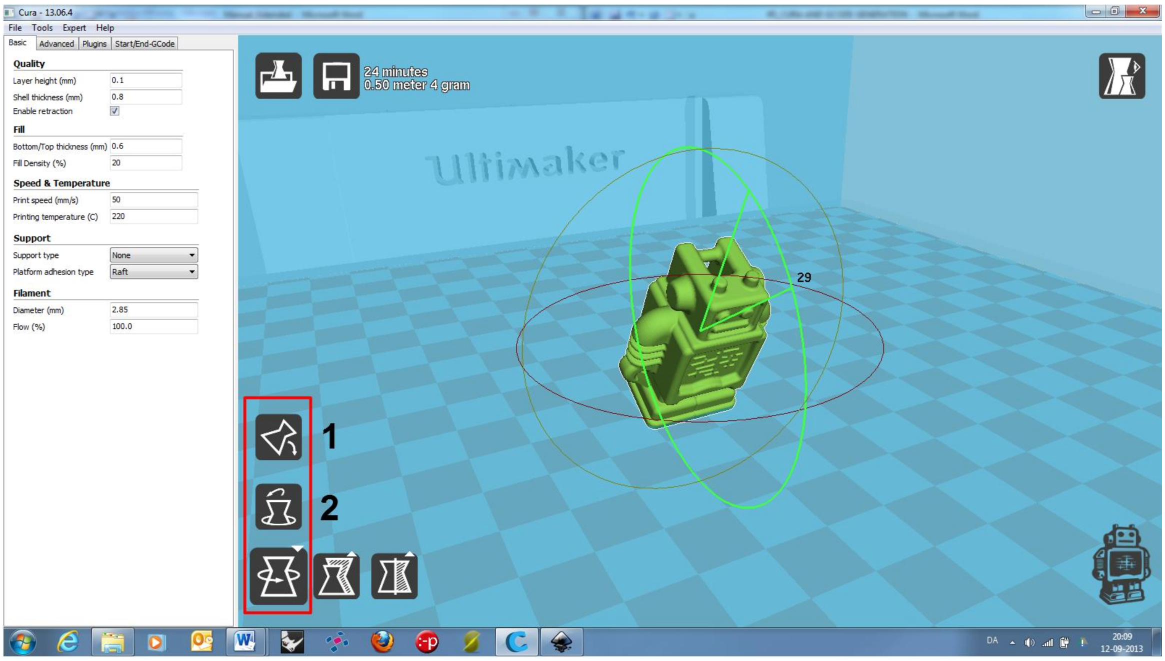


MANIPULATION TOOLS

As mentioned previously Cura has 3 manipulation tools that allow the adjustment of the part directly on the software instead to go back to the CAD software. These manipulation tools are:

- Rotation
- Scaling
- Mirroring

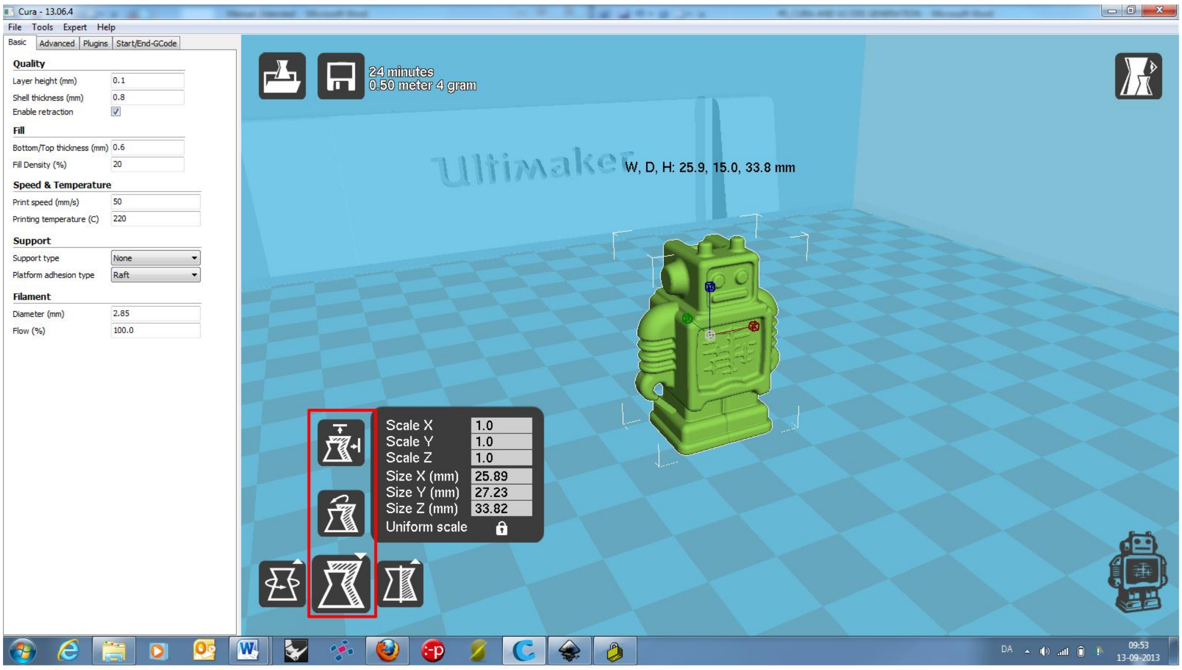
ROTATION



When selecting the *Rotation* tool three colored axis will appear around the object. By selecting one of them at the time it is possible to change the orientation of the part along the three Cartesian axes. When moving the circles, a number with the number of degrees is displayed.

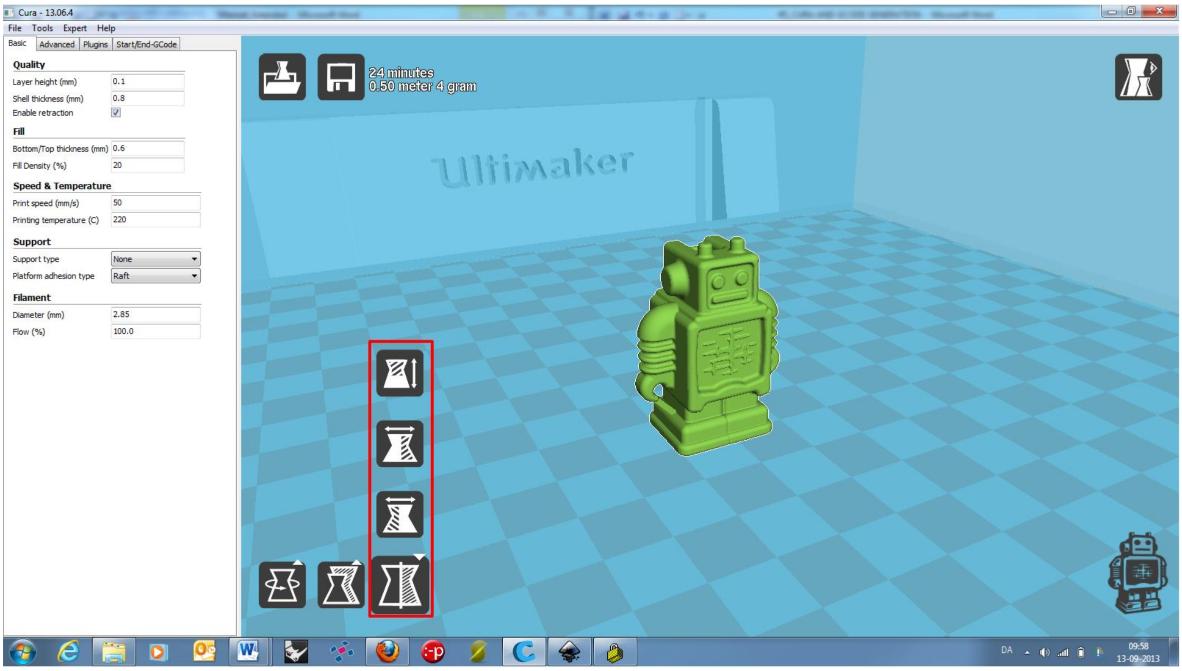
When selecting the rotation option two other icons will appear. They represent the *Lay flat* (nr. 1 in the picture above) and *Reset* (nr. 2 in the picture above) options.

SCALE



It is possible to scale the parts by clicking the rotation tool and then setting the required scaling values on the box or doing a click and move on the little boxes over the model. The overall part dimensions wanted can also be set, and the scaling values will be calculated automatically.

MIRROR



It is also possible to mirror the object directly in Cura by selecting the *Mirror* option in the lower left corner.

SAVING THE FILES ON THE SD CARD

When making some changes on the settings or in the viewport (i.e. scaling or mirroring), the GCode is automatically generated. The progress of the generation of the Gcode is visible on the top left *Save* icon. Whenever Cura recalculate the paths a new information on the printing time and estimated material used is also shown.

To save the Gcode on the SD card: *File> Save GCode > select destination (SD card)> Save*.